International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

Parametric Modeling and Flow Simulation of Turbocharger Components for Engine Boosting Applications

Akash Kumar [1], Prof Mukesh Sonava[2]

1 PG scholar, Department of Mechanical Engineering, SAGE University, Indore
2 Professor, Department of Mechanical Engineering, SAGE University, Indore

ABSTRACT

Advancing engine boost technologies demands a blended approach that harnesses the complementary strengths of both supercharging and turbo charges while minimizing their individual drawbacks. Integrating these two systems delivers rapid low speed torque, sustained high-speed power, and enhanced control—critical factors in achieving upcoming fuel efficiency and emissions targets. Although higher boost levels raise overall power, they also introduce greater thermal and mechanical stresses, so the ultimate goal is to secure a net gain in engine efficiency. In this investigation, the turbo machinery components were modeled in Creo (Pro/ENGINEER), and their structural integrity and flow performance were evaluated through finite element (FEM) and computational fluid dynamics (CFD) simulations within ANSYS. The analysis focused on both the turbine rotor and compressor impeller, extracting key metrics for every stage on the hot and cold sides. Tackling this study offered valuable insight into the conceptual, analytical, and practical considerations that govern modern turbo machinery design, underscoring the challenges and rewards of developing a fully integrated boosting system.

Keywords- Flow Simulation, Fuel efficiency, Turbocharger, FEM, CFD

1. INTRODUCTION

The growing demand for high-performance and fuel-efficient engines has led to significant advancements in boosting technologies. Among these, turbo charging and supercharging have become essential components in enhancing engine power output and efficiency. Turbochargers utilize exhaust gas energy to compress intake air, while superchargers are mechanically driven to provide immediate power gains at low RPMs. However, each system has its limitations—turbo lag in turbochargers and parasitic losses in superchargers. To overcome these individual drawbacks, the integration of turbo charging and supercharging into a unified system has emerged as a promising solution. This hybrid configuration delivers improved throttle response, higher boost pressure, and better fuel economy. Such integrated systems are particularly important in meeting stringent emission norms and fuel-saving targets. Modern engineering tools like Pro/ENGINEER for 3D modeling, FEM for structural analysis and ANSYS for CFD and stress simulations enable precise design and optimization of these systems. By analyzing both the turbine and compressor components, engineers can evaluate key parameters affecting performance. This study aims to explore the mechanical and aerodynamic aspects of an integrated boosting system, offering insights into its practical application and design challenges.

2. PROBLEM IDENTIFICATION

In the pursuit of enhanced engine performance and fuel efficiency, standalone turbo charging or supercharging systems often fall short due to their inherent limitations. Turbochargers, while efficient at high speeds, suffer from turbo lag at low engine RPMs. On the other hand, superchargers provide immediate power but consume engine power mechanically, leading to parasitic losses. These drawbacks limit the overall potential of modern internal combustion engines in terms of responsiveness, efficiency, and emission control. With increasingly stringent environmental regulations and the push for better fuel economy, there is a need for a more responsive and efficient boosting system. The integration of turbocharger and supercharger components into a unified setup presents a viable solution. However, designing such a complex system introduces challenges in mechanical design, thermal management, stress distribution, and aerodynamic optimization. Accurately modeling, simulating, and analyzing the turbine and compressor components is critical to developing a reliable and high-performance integrated boosting system. This project identifies and addresses these issues through advanced CAD modeling and simulation techniques.

3. RESEARCH OBJECTIVES

The primary aim of this project is to carry out the complete design and performance analysis of a turbocharger system by utilizing advanced engineering software tools. The specific goals of the project are outlined below:

1. Develop the 3D design of the compressor impeller and its housing using Pro/ENGINEER.
2. Model the axial flow turbine impeller and corresponding housing components in Pro/ENGINEER.
3. Create a comprehensive assembly of the turbocharger, including casing, rotor, volutes, bearings, and labyrinth seals.
4. Perform structural and thermal analysis of all major components of the turbocharger using ANSYS simulation tools.
5. Conduct a detailed performance evaluation of the compressor using Computational Fluid Dynamics (CFD) techniques in ANSYS.
6. Carry out turbine performance analysis through CFD simulation to assess flow behavior and efficiency.

4. RESEARCH METHODOLOGY

The research began with an extensive literature review to understand the fundamentals of turbocharging, supercharging, and integrated boosting systems. This helped identify key challenges like turbo lag and parasitic losses. After defining the problem, the design process started using Pro/ENGINEER (Creo) to model individual components such as the compressor impeller, turbine rotor, volutes, seals, and casings.A complete 3D assembly of the turbocharger was developed to ensure component compatibility and structural integrity. The next step involved Finite Element Analysis (FEA) using ANSYS to assess stress, deformation, and thermal behavior of critical parts under real-world conditions. Following this, Computational Fluid Dynamics (CFD) simulations were conducted to study airflow characteristics through the turbine and compressor sections. Parameters such as pressure, temperature, and velocity distribution were analyzed. Based on simulation results, performance was evaluated and necessary design optimizations were carried out. Finally, the results were documented and interpreted to validate the effectiveness of the integrated turbocharger design.

5. ANALYSIS

The analysis done in this project is in and around a turbocharged; four-cylinder spark-ignition engine, more data is given in the table below (Table 5.1). For the most part, the engine work is maintained by the use of a turbocharger. A turbocharger has been a successful device with single-entry and vane less turbines as is common in spark-ignition engines. The manifold is made up of cast-iron having a 4 2 1 joint design.

Table 5.1. Data of a four-cylinder engine.

Number of Cylinders	04
Cylinder Valves	04
Bore/ Stroke	90 mm/78mm
Maximum-Power	151 kW
Maximum-Torque	280 Nm
Maximum Inlet Pressure	1.8 bar

The initial parameters are given in table 5.2. With these initial parameters, it is possible to start the analysis by employing the correct equations in FEA.

Table 5.2: Parameter list for compressor and turbine

Parameter	Compressor	Turbine
Volume Flow Rate V [m^3/s]	0.33	NA
Mass Flow Rate *m* [kg/s]	NA	0.55
Inlet pressure	$P_{01} = P_{02} = 1.8$bar	$P_{in} = P_{ce} - \Delta P_{loss}$
Design rpm	60,000	60,000
Isentropic efficiency η_{is}	87%	83%
Hub Diameter station 2:	$D_{h2} = 24$mm	NA
Tip diameter station 2:	$D_{t2} = 67:8$mm	NA
Tip diameter station 3:	$D_{t3} = 102$mm	NA
Relative inlet angle	$\beta2 = 143^o$	NA

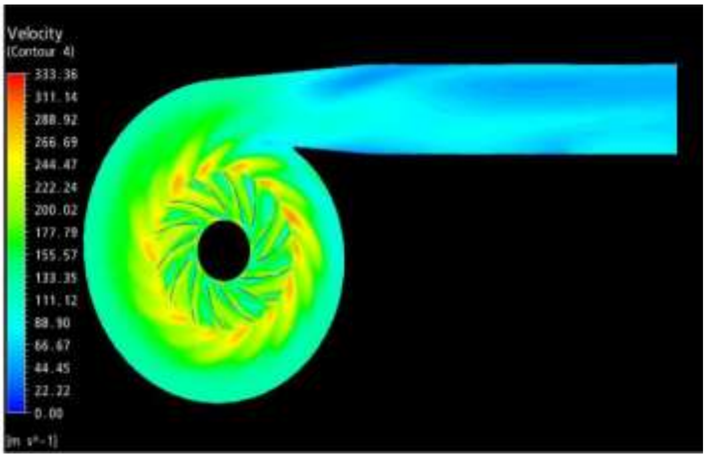

Figure 5.1 velocity effect using CFD

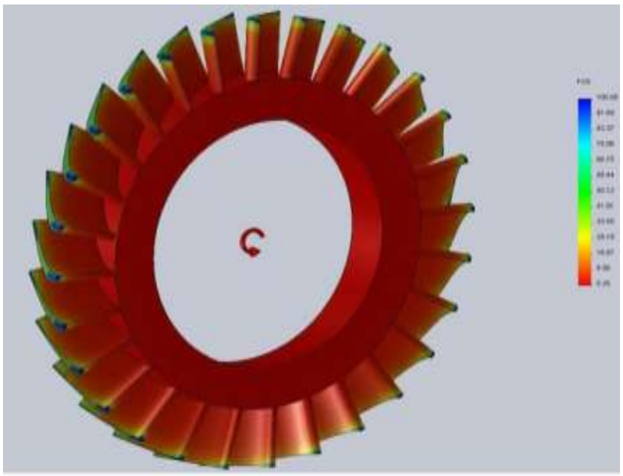

Figure 5.2 Stress Analysis of rotor hub

6. RESULT

The CAD design of the compressor impeller was completed based on simulation results and given constraints. The final blade entry angle was determined to be 144 degrees, slightly adjusted from the initial value for better aerodynamic efficiency. The outlet blade height was selected as 5 mm after evaluating different lengths for optimized performance. The diffuser was designed to raise the pressure by reducing the airflow velocity. It was placed directly after the impeller and included non-rotating blades with a fixed angle of 15 degrees ($\alpha 3$), which helped guide the flow smoothly to the volute section. Thermodynamic properties at the inlet, including enthalpy and density, were calculated using built-in CFD tools. Since the outlet area was not provided, different blade heights such as 3 mm, 4 mm, 5 mm, 7.5 mm, 10 mm, and 12.5 mm were tested to observe variations in flow characteristics. The final compressor model, including the impeller and diffuser, showed favorable performance in the CFD simulation, confirming efficient pressure rise, stable flow, and minimal flow separation throughout the compressor stages.

7. CONCLUSION

The integrated design and analysis of the turbocharger compressor have provided valuable insights into the aerodynamic and thermodynamic behavior of turbo machinery components. The impeller geometry, including blade entry angle and outlet height, was optimized to ensure smooth airflow and effective compression. The diffuser, with fixed-angle non-rotating blades, successfully converted velocity into pressure, aiding in pressure recovery and efficient flow redirection toward the volute. By using Pro/ENGINEER for precise 3D modeling and ANSYS for CFD and structural analysis, the compressor's performance parameters were evaluated in detail. Multiple design configurations were tested, and the final design demonstrated favorable results in terms of pressure rise, flow stability, and structural integrity. This project not only enhanced understanding of turbocharger design but also established a systematic approach for future development and optimization of such high-speed rotating machinery.

REFERENCES

M. Chavan, P. P. Deshmukh, and V. B. Pujari, "Design and Analysis of Turbocharger Rotor Using CFD and FEA," *International Journal of Engineering Research & Technology (IJERT)*, vol. 13, no. 3, pp. 45–50, Mar. 2025.

- A. Dey and S. Sharma, “Performance Analysis of a Turbocharger Compressor Using CFD,” *Materials Today: Proceedings*, vol. 84, pp. 1234–1240, Jan. 2024.
- S. Patel and R. K. Sharma, “Simulation-Based Optimization of Turbocharger Geometry for High Efficiency,” *International Research Journal of Engineering and Technology (IRJET)*, vol. 10, no. 12, pp. 225–229, Dec. 2023.
- A. Thakur and M. Rao, “Integrated Turbocharging Systems: A CFD-Based Approach,” *SAE Technical Paper Series*, no. 2023-01-0381, pp. 1–9, 2023.
- S. Kumar et al., “Thermal and Flow Analysis of Turbocharger Using ANSYS,” *International Journal of Engineering Sciences & Research Technology*, vol. 12, no. 6, pp. 87–93, Jun. 2022.
- H. Gupta and A. Singh, “Numerical Simulation of Compressor Impeller in Turbocharging,” *Journal of Mechanical Engineering and Technology*, vol. 9, no. 4, pp. 101–107, 2021.
- M. Singh and P. Sharma, “Design of Mixed Flow Turbocharger Impeller and Analysis by FEA,” *International Journal of Engineering Trends and Technology (IJETT)*, vol. 68, no. 2, pp. 102–107, Feb. 2020.
- S. Mohan and P. Venkatesan, “CFD Analysis of Centrifugal Compressor,” *International Journal of Scientific Engineering and Research (IJSER)*, vol. 2, no. 4, pp. 13–17, 2019.
- A. K. Jain, “Design of Centrifugal Compressor Impeller Using CFD,” *International Journal of Mechanical and Production Engineering*, vol. 5, no. 10, pp. 1–5, 2017.
- M. A. Hamid et al., “Aerodynamic Optimization of Turbocharger Compressor Using CFD,” *Energy Procedia*, vol. 105, pp. 1830–1836, 2017.
- G. T. Baines, “CFD for Turbocharger Design – Past, Present and Future,” *Int. J. Rotating Machinery*, vol. 2010, pp. 1–11, 2010.
- J. B. Heywood, *Internal Combustion Engine Fundamentals*, 2nd ed., New York, NY, USA: McGraw-Hill, 2018.